CHAPTER 4

MACHINE DRAWING

When you have read and understood this chapter, you should be able to answer the following learning objectives:

- *Describe basic machine drawings.*
- *Describe the types of machine threads.*
- *Describe gear and helical spring nomenclature.*
- *Explain the use of finish marks on drawings.*

This chapter discusses the common terms, tools, and conventions used in the production of machine drawings.

COMMON TERMS AND SYMBOLS

In learning to read machine drawings, you must first become familiar with the common terms, symbols, and conventions defined and discussed in the following paragraphs.

GENERAL TERMS

The following paragraphs cover the common terms most used in all aspects of machine drawings.

Tolerances

Engineers realize that absolute accuracy is impossible, so they figure how much variation is permissible. This allowance is known as tolerance. It is stated on a drawing as (plus or minus) a certain amount, either by a fraction or decimal. Limits are the maximum and/or minimum values prescribed for a specific dimension, while tolerance represents the total amount by which a specific dimension may vary. Tolerances may be shown on drawings by several different methods; figure 4-1 shows three examples. The unilateral method (view A) is used when variation from the design size is permissible in one direction only. In the bilateral method (view B), the dimension figure shows the plus or minus variation that is acceptable. In the limit dimensioning method (view C), the maximum and minimum measurements are both stated

The surfaces being toleranced have geometrical characteristics such as roundness, or perpendicularity to another surface. Figure 4-2 shows typical geometrical characteristic symbols. A datum is a surface, line, or

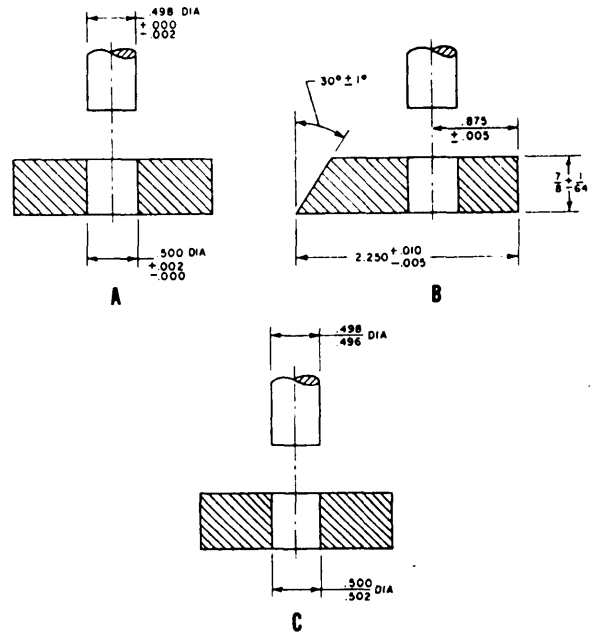

Figure 4-1.—Methods of indicating tolerance.

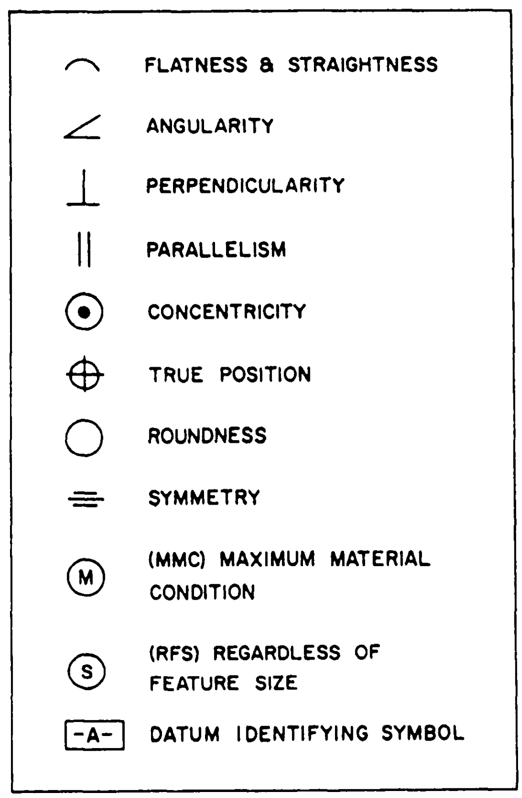

Figure 4-2.—Geometric characteristic symbols.

point from which a geometric position is to be determined or from which a distance is to be measured. Any letter of the alphabet except I, O, and Q may be used as a datum identifying symbol. A feature control symbol is made of geometric symbols and tolerances. Figure 4-3 shows how a feature control symbol may include datum references.

Fillets and Rounds

Fillets are concave metal corner (inside) surfaces. In a cast, a fillet normally increases the strength of a metal corner because a rounded corner cools more evenly than a sharp corner, thereby reducing the possibility of a break. Rounds or radii are edges or outside corners that have been rounded to prevent chipping and to avoid sharp cutting edges. Figure 4-4 shows fillets and rounds.

Slots and Slides

Slots and slides are used to mate two specially shaped pieces of material and securely hold them together, yet allow them to move or slide. Figure 4-5 shows two types: the tee slot, and the dovetail slot. For examples, a tee slot arrangement is used on a milling machine table, and a dovetail is used on the cross slide assembly of an engine lathe.

Keys, Keyseats, and Keyways

A key is a small wedge or rectangular piece of metal inserted in a slot or groove between a shaft and a hub to prevent slippage. Figure 4-6 shows three types of keys.

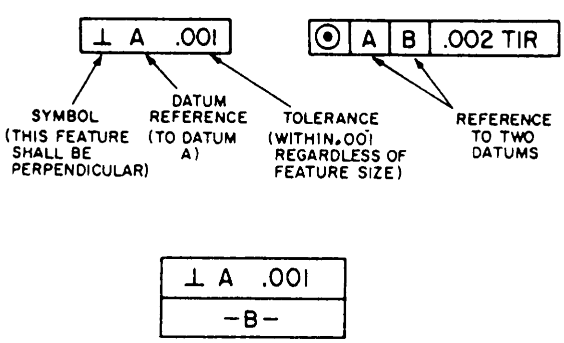

Figure 4-3.—Feature control frame indicating a datum reference.

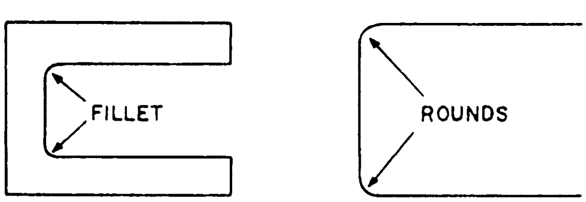

Figure 4-4.—Fillets and rounds

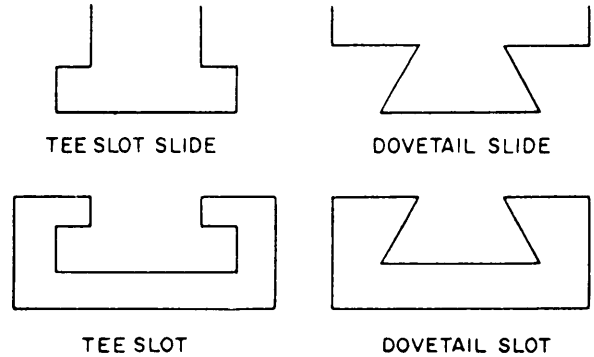

Figure 4-5.—Slots and slides.

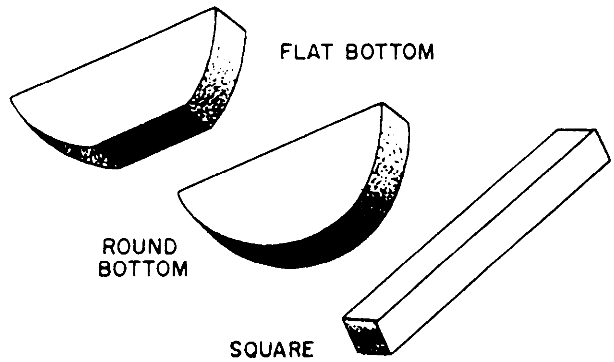

Figure 4-6.—Three types of keys.

Figure 4-7 shows a keyseat and keyway. View A shows a keyseat, which is a slot or groove on the outside of a part into which the key fits. View B shows a keyway, which is a slot or groove within a cylinder, tube, or pipe. A key fitted into a keyseat will slide into the keyway and prevent movement of the parts.

SCREW THREADS

Draftsmen use different methods to show thread on drawings. Figures 4-8 through 4-11 show several of

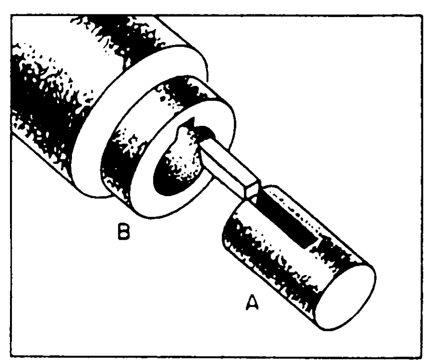

Figure 4-7.—A keyseat and keyway.

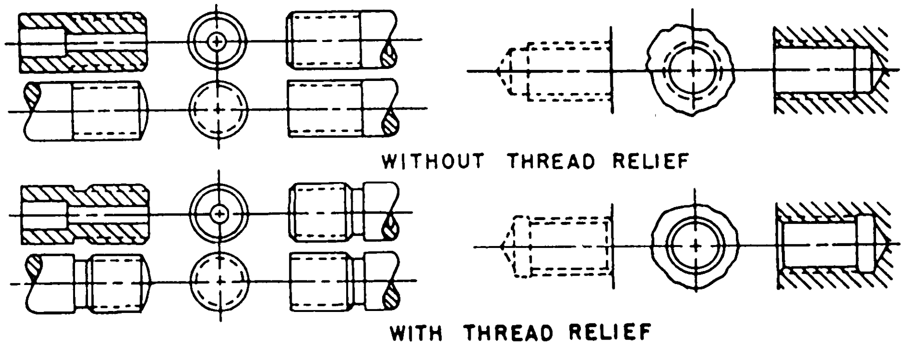

Figure 4-8.—Simplified method of thread representation.

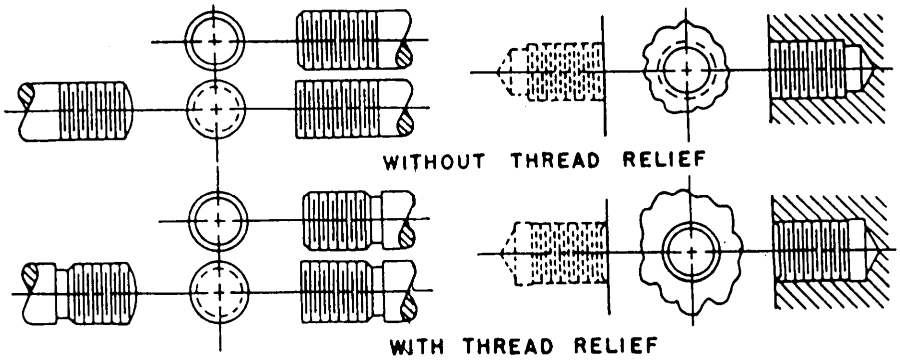

Figure 4-9.—Schematic method of thread representation.

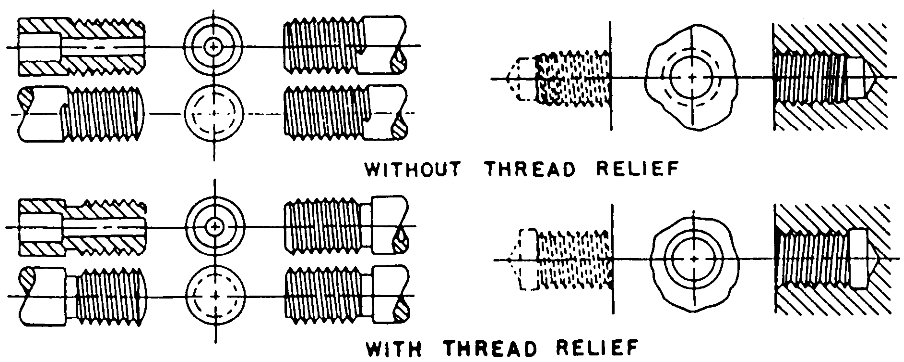

Figure 4-10.—Detailed method of thread representation.

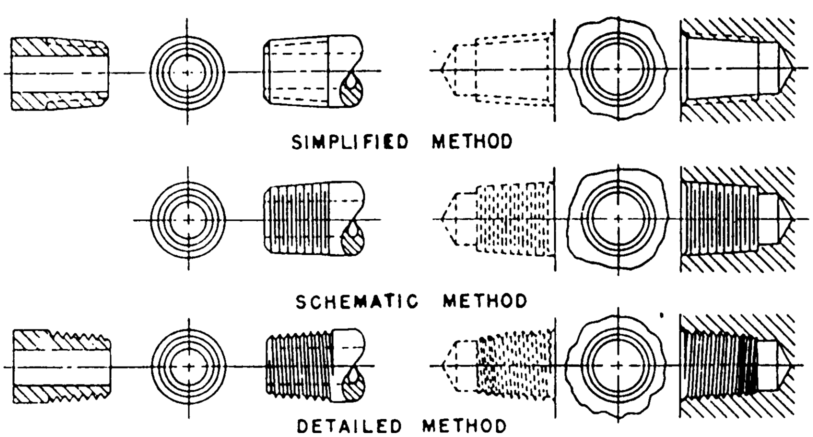

FIgure 4-11.—Tapered pipe thread representation.

them. Now look at figure 4-12. The left side shows a thread profile in section and the right side shows a common method of drawing threads. To save time, the draftsman uses symbols that are not drawn to scale. The drawing shows the dimensions of the threaded part but other information may be placed in "notes" almost any place on the drawing but most often in the upper left comer. However, in our example the note is directly above the drawing and shows the thread designator "1/4-20 UNC-2."

The first number of the note, 1/4, is the nominal size which is the outside diameter. The number after the first dash, 20, means there are 20 threads per inch The letters UNC identify the thread series as Unified National Coarse. The last number, 2, identifies the class of thread and tolerance, commonly called the fit. If it is a left-hand thread, a dash and the letters LH will follow the class of thread. Threads without the LH are right-hand threads.

Specifications necessary for the manufacture of screws include thread diameter, number of threads per inch, thread series, and class of thread The two most widely used screw-thread series are (1) Unified or National Form Threads, which are called National Coarse, or NC, and (2) National Fine, or NF threads. The NF threads have more threads per inch of screw length than the NC.

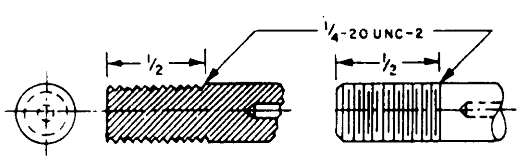

Figure 4-12.—Outside threads.

Classes of threads are distinguished from each other by the amount of tolerance and/or allowance specified Classes of thread were formerly known as *class of fit,* a term that will probably remain in use for many years. The new term, *class of thread,* was established by the National Bureau of Standards in the *Screw-Thread Standards for Federal Services,* Handbook H-28.

Figure 4-13 shows the terminology used to describe screw threads. Each of the terms is explained in the following list:

HELIX—The curve formed on any cylinder by a straight line in a plane that is wrapped around the cylinder with a forward progression.

EXTERNAL THREAD—A thread on the outside of a member. An example is the thread of a bolt.

INTERNAL THREAD—A thread on the inside of a member. An example is the thread inside a nut.

MAJOR DIAMETER—The largest diameter of an external or internal thread

AXIS—The center line running lengthwise through a screw.

CREST—The surface of the thread corresponding to the major diameter of an external thread and the minor diameter of an internal thread.

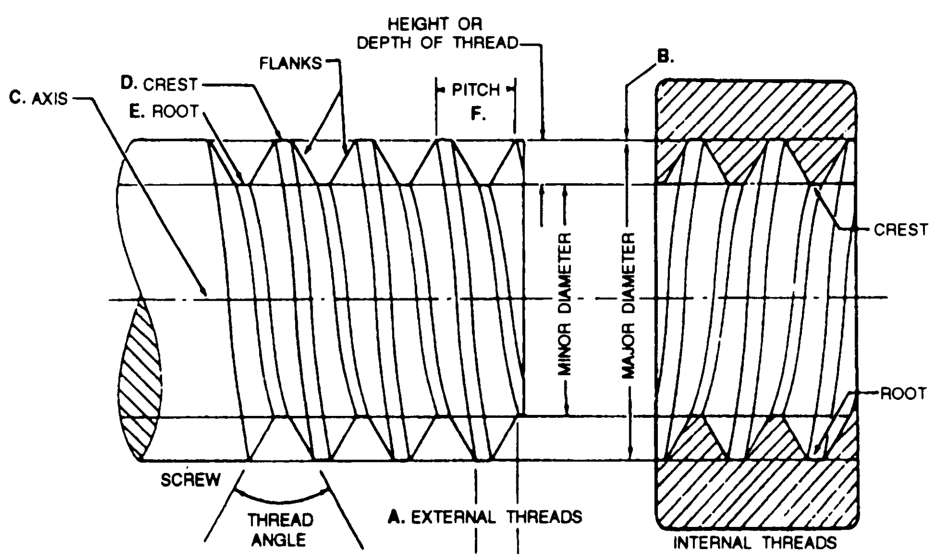

Figure 4-13.—Screw thread terminology.

ROOT—The surface of the thread corresponding to the minor diameter of an external thread and the major diameter of an internal thread

DEPTH—The distance from the root of a thread to the crest, measured perpendicularly to the axis.

PITCH—The distance from a point on a screw thread to a corresponding point on the next thread, measured parallel to the axis.

LEAD—The distance a screw thread advances on one turn, measured parallel to the axis. On a single-thread screw the lead and the pitch are identical; on a double-thread screw the lead is twice the pitch; on a triple-thread screw the lead is three times the pitch

GEARS

When gears are drawn on machine drawings, the draftsman usually draws only enough gear teeth to identify the necessary dimensions. Figure 4-14 shows gear nomenclature, and the terms in the figure are explained in the following list:

PITCH DIAMETER (PD)—The diameter of the pitch circle (or line), which equals the number of teeth on the gear divided by the diametral pitch

DIAMETRAL PITCH (DP)—The number of teeth to each inch of the pitch diameter or the number of teeth on the gear divided by the pitch diameter. Diametral pitch is usually referred to as simply PITCH.

NUMBER OF TEETH (N)—The diametral pitch multiplied by the diameter of the pitch circle (DP x PD).

ADDENDUM CIRCLE (AC)—The circle over the tops of the teeth.

OUTSIDE DIAMETER (OD)—The diameter of the addendum circle.

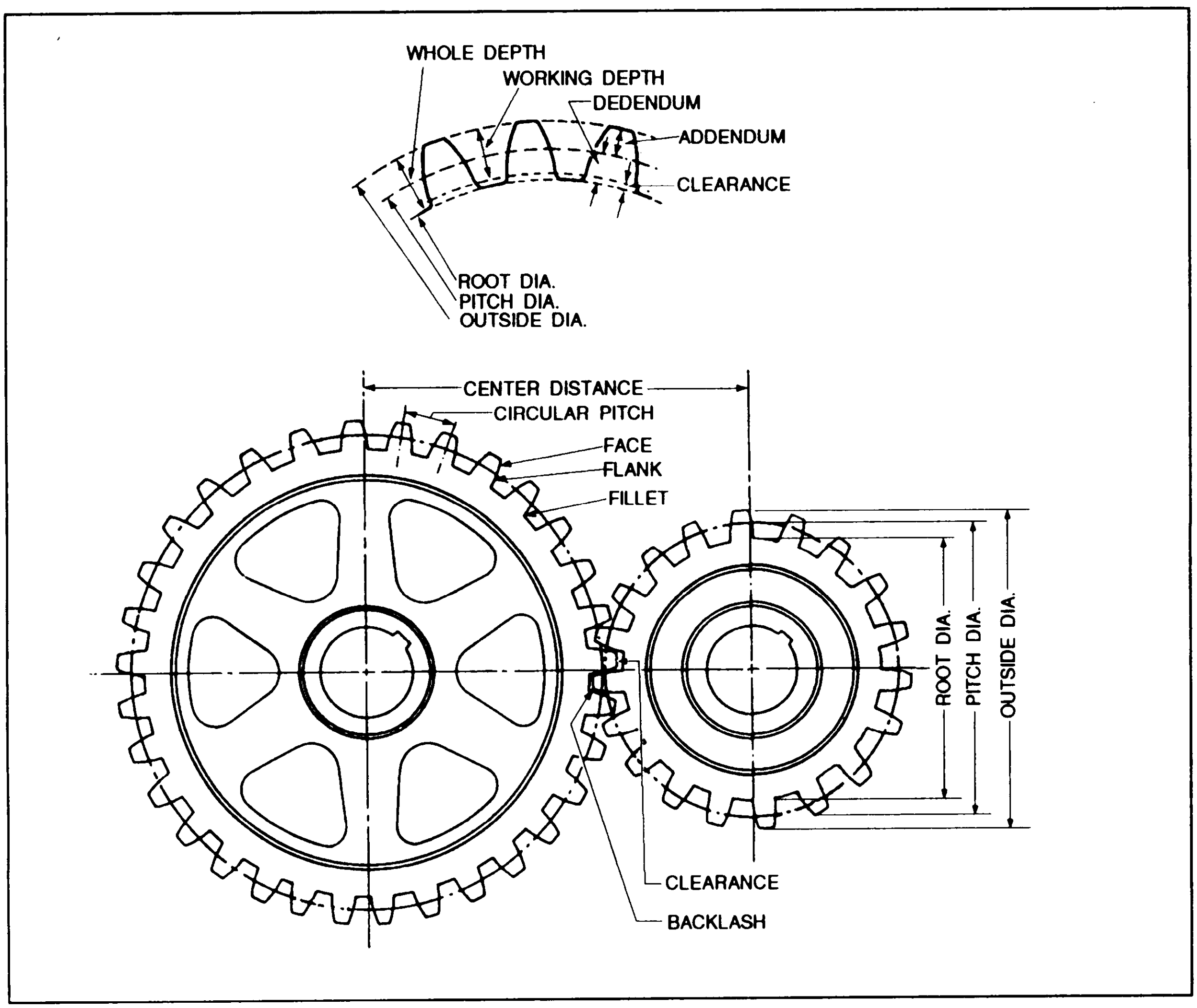

Figure 4-14.—Gear nomenclature.

CIRCULAR PITCH (CP)—The length of the arc of the pitch circle between the centers or corresponding points of adjacent teeth.

ADDENDUM (A)—The height of the tooth above the pitch circle or the radial distance between the pitch circle and the top of the tooth.

DEDENDUM (D)—The length of the portion of the tooth from the pitch circle to the base of the tooth.

CHORDAL PITCH—The distance from center to center of teeth measured along a straight line or chord of the pitch circle.

ROOT DIAMETER (RD)—The diameter of the circle at the root of the teeth.

CLEARANCE (C)—The distance between the bottom of a tooth and the top of a mating tooth.

WHOLE DEPTH (WD)—The distance from the top of the tooth to the bottom, including the clearance.

FACE—The working surface of the tooth over the pitch line.

THICKNESS—The width of the tooth, taken as a chord of the pitch circle.

PITCH CIRCLE—The circle having the pitch diameter.

WORKING DEPTH—The greatest depth to which a tooth of one gear extends into the tooth space of another gear.

RACK TEETH—A rack may be compared to a spur gear that has been straightened out. The linear pitch of the rack teeth must equal the circular pitch of the mating gear.

HELICAL SPRINGS

There are three classifications of helical springs: compression, extension, and torsion. Drawings seldom show a true presentation of the helical shape; instead, they usually show springs with straight lines. Figure 4-15 shows several methods of spring representation including both helical and straight-line drawings. Also, springs are sometimes shown as single-line drawings as in figure 4-16.

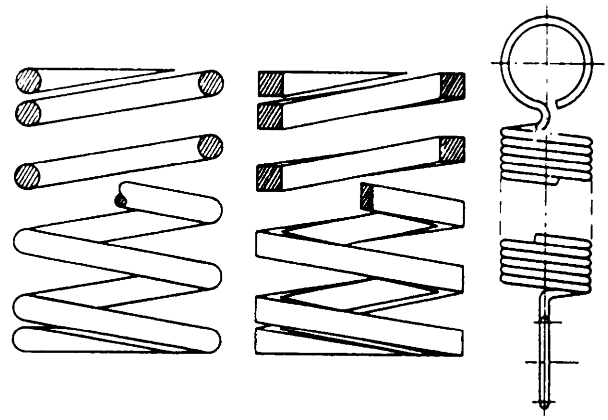

Figure 4-15.—Representation of commm types of helical springs.

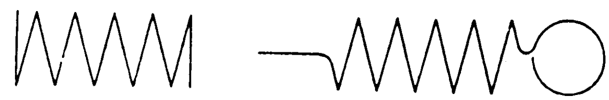

Figure 4-16.—Single line representation of springs

FINISH MARKS

The military standards for finish marks are set forth in ANSI 46.1-1962. Many metal surfaces must be finished with machine tools for various reasons. The acceptable roughness of a surface depends upon how the part will be used. Sometimes only certain surfaces of a part need to be finished while others are not. A modified symbol (check mark) with a number or numbers above it is used to show these surfaces and to specify the degree of finish. The proportions of the surface roughness symbol are shown in figure 4-17. On small drawings the symbol is proportionately smaller.

The number in the angle of the check mark, in this case 02, tells the machinist what degree of finish the surface should have. This number is the root-mean-square value of the surface roughness height in millionths of an inch. In other words, it is a measurement of the depth of the scratches made by the machining or abrading process.

Wherever possible, the surface roughness symbol is drawn touching the line representing the surface to

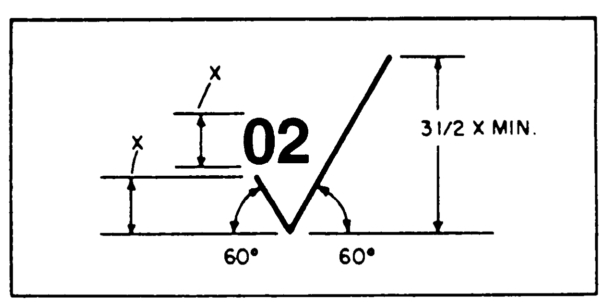

Figure 4-17.—Proportions for a basic finish symbol.

which it refers. If space is limited, the symbol may be placed on an extension line on that surface or on the tail of a leader with an arrow touching that surface as shown in figure 4-18.

When a part is to be finished to the same roughness all over, a note on the drawing will include the direction "finish all over" along the finish mark and the proper number. An example is FINISH ALL OVER$\overset{32}{\surd}$. When a part is to be finished all over but a few surfaces vary in roughness, the surface roughness symbol number or numbers are applied to the lines representing these surfaces and a note on the drawing will include the surface roughness symbol for the rest of the surfaces. For example, ALL OVER EXCEPT AS NOTED (fig. 4-19).

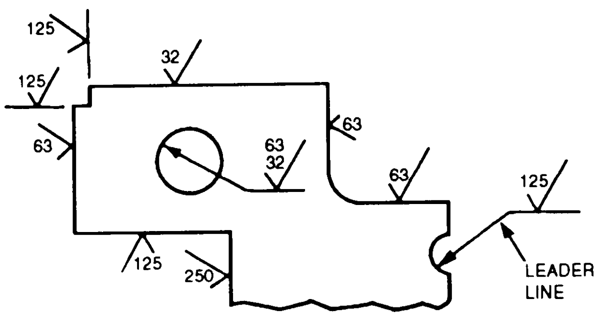

Figure 4-18.—Methods of placing surface roughness symbols.

STANDARDS

American industry has adopted a new standard, *Geometrical Dimensioning and Tolerancing,* ANSI Y14.5M-1982. This standard is used in all blueprint production whether the print is drawn by a human hand or by computer-aided drawing (CAD) equipment. It standardizes the production of prints from the simplist hand-made job on site to single or multiple-run items produced in a machine shop with computer-aided manufacturing (CAM) which we explained in chapter 2. DOD is now adopting this standard For further information, refer to ANSI Y14.5M-1982 and to *Introduction to Geometrical Dimensioning and Tolerancing,* Lowell W. Foster, National Tooling and Machining Association, Fort Washington, MD, 1986.

The following military standards contain most of the information on symbols, conventions, tolerances, and abbreviations used in shop or working drawings:

ANSI Y14.5M-1982	Dimensioning and Tolerancing
MIL-STD-9A	Screw Thread Conventions and Methods of Specifying
ANSI 46.1	Surface Texture
MIL-STD-12,C	Abbreviations for Use On Drawings and In Technical-Type Publications

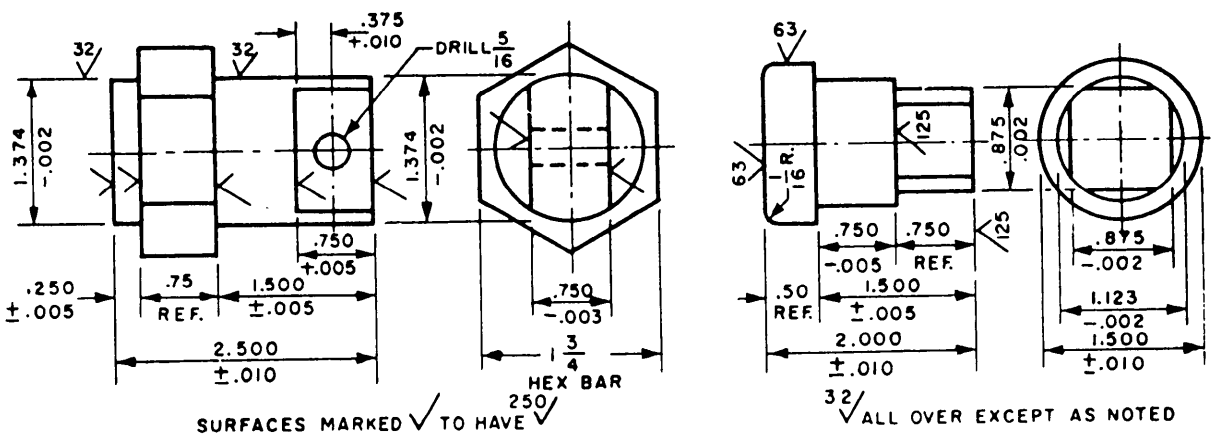

FIgure 4-19.—Typical examples of symbol use.